

## **Computational Fluid Dynamics Modelling to design and optimise Power Kites for Renewable Power Generation**

SURI, Yatin, PEGG, C, ISLAM, S and ASTHANA, Abhishek

Available from Sheffield Hallam University Research Archive (SHURA) at:

<http://shura.shu.ac.uk/26025/>

---

This document is the author deposited version. You are advised to consult the publisher's version if you wish to cite from it.

### **Published version**

SURI, Yatin, PEGG, C, ISLAM, S and ASTHANA, Abhishek (2019). Computational Fluid Dynamics Modelling to design and optimise Power Kites for Renewable Power Generation. In: AL-HABIBEH, Amin, ASTHANA, Abhishek and VUKOVIC, Vladimir, (eds.) The International Conference on Energy and Sustainable Futures (ICESF). Nottingham Trent University Publications.

---

### **Copyright and re-use policy**

See <http://shura.shu.ac.uk/information.html>



## Computational Fluid Dynamics Modelling to design and optimise Power Kites for Renewable Power Generation

Yatin Suri <sup>1</sup>, Christopher Pegg <sup>1</sup>, Sheikh Zahidul Islam <sup>1</sup>, Abhishek Asthana <sup>2</sup>

<sup>1</sup> *The Robert Gordon University, School of Engineering, Aberdeen, U.K.*

<sup>2</sup> *Sheffield Hallam University, Hallam Energy, Sheffield, U.K*

[y.suri@rgu.ac.uk](mailto:y.suri@rgu.ac.uk)

### Abstract

Power kites provide the potential rewards of obtaining the disused energy supply from high altitude wind. This paper aims to provide a design of Power kite and optimise the potential for renewable power generation. The Power kite was modelled using Computational Fluid Dynamics (CFD) to study its characteristics. The numerical modelling results were compared against the wind tunnel experimental study and two 3D printed Power kites. The design was optimised using several variables, including aerofoil choice, surface roughness, wind speed and operating parameters. Attempts at optimising the kite design were implemented. The results suggest that operating the kites at minimum 15 m horizontal separation is favourable, with the trailing kite operating below the leading, removing the potential for this kite to operate in the wake turbulence of the first. As the wind speed is generally very low at low altitudes, it is recommended to use a symmetrical aerofoil for the kite design, as these tend to produce greater lift with low Reynolds airflow. This paper presents relevant, applicable data which can be used for predicting the performance, and potentially optimising further Power kite designs.

**Keywords:** Power kites, CFD, Wind energy, Renewable power generation

### 1.Introduction

Wind energy is the world's fastest growing renewable resource sector, and in many countries, is the largest provider of renewable power [1]. Unfortunately, wind turbines have inherent limitations when harnessing the power of the wind. It is well recognised that with increased altitude come increased wind power densities; this is reflected in the continual rise in wind turbine heights [2]. The current wind turbine technology possesses limitation to harness the

abundant natural resource of high altitude wind [3]. Power kites are more effective at high altitudes due to both aerofoil design and location of components [4], the generator of a power kite is left on the ground; connected by a tether. This allows the kite to spin the generator, as the available power at high altitudes is significantly greater [2].

Being a somewhat novel approach, there exist few readily available resources to predict kite performance accurately [2,3]. Without accurate performance prediction and the ability to optimise kite design at an early stage, these power generation systems will struggle to reach their full potential [5]. Thus, it is important to maximise the lift produced to optimise the efficiency. Previous research [2,3,5] has targeted aerofoil shape which is undoubtedly a significant factor in efficiency but surface roughness was ignored in these studies. An aerofoil with a high surface roughness will produce turbulent airflow over the body, which, in turn, will increase the resistance due to drag on the kite [3]. Drag also occurs on the tether over its length; over 500 metres it is possible to develop significant drag, enough to have a large impact on the overall efficiency of the system. It is, therefore, necessary to study aerodynamic drag over the kite, while also optimising the aerofoil shape to produce maximum lift [2,3]. This paper uses computational fluid dynamics to solve Navier-Stokes equations and model the wind flow in design and optimisation of power kites, under varying input parameters, namely horizontal and vertical separation distances and surface roughness. The role of the change in lift and drag coefficients has been investigated in detail.

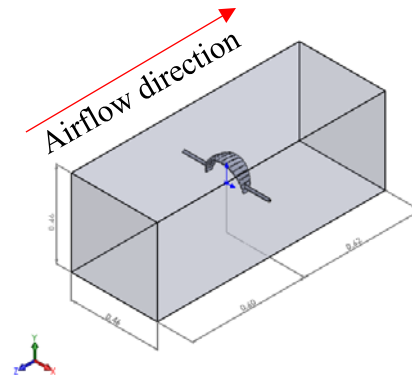
### **Computational domain and Boundary Conditions**

In the present study, NACA 0018 Aerofoil is used [4]. The kite dimensions can be found in Table 1. The computational domain used in the present study is in the form of a cuboid with dimensions 1.22 m×0.46 m×0.46 m based on the wind tunnel experiment performed shown in Figure 1. The kite is located at 0.6 m from the inlet. The experiment and the validation are performed with a single kite. Following that two kites are located and analysis was carried out to understand the horizontal and vertical separation distance between them. The Power kite was modelled using Computational Fluid Dynamics (CFD) by solving the Navier-Stokes equation (continuity and momentum conservation equations) assuming isothermal, steady state, and incompressible flow. A velocity inlet boundary condition was used with 10 m/s at inlet and pressure outlet was used at the outlet with 0 Pa pressure. All the other boundaries were configured as no slip wall. A mesh independence study was performed by progressively refining the mesh whilst monitoring the lift ( $C_L$ ) and drag ( $C_D$ ) coefficients. Convergence was

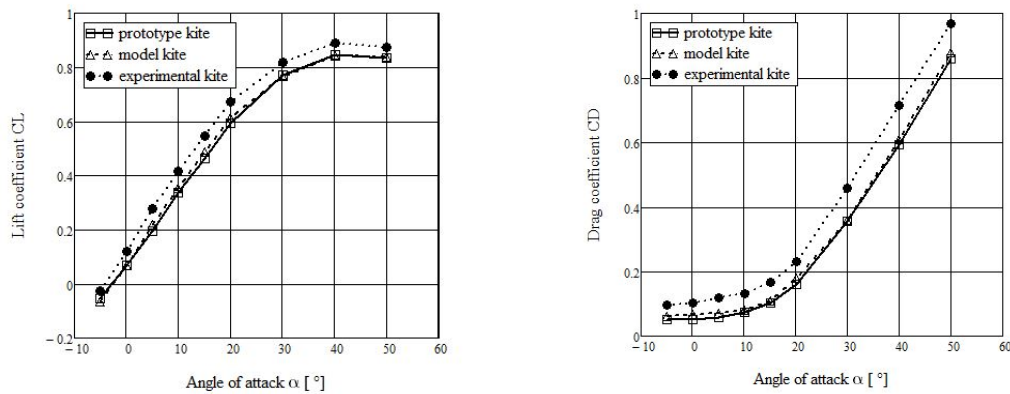
deemed to have been reached when refining the mesh began to have little to no effect on both these coefficients. The number of cells used in the present study are  $8 \times 10^5$ .

*Table 1: Kite dimensions*

| Component       | Length<br>(m) | Angle<br>(°) |
|-----------------|---------------|--------------|
| Wingspan        | 11.128        | N/A          |
| Centre<br>Chord | 2.903         | N/A          |
| End Chord       | 1.7418        | N/A          |
| Kite Arc        | N/A           | 180          |



*Figure 1: Computational domain, in meters*



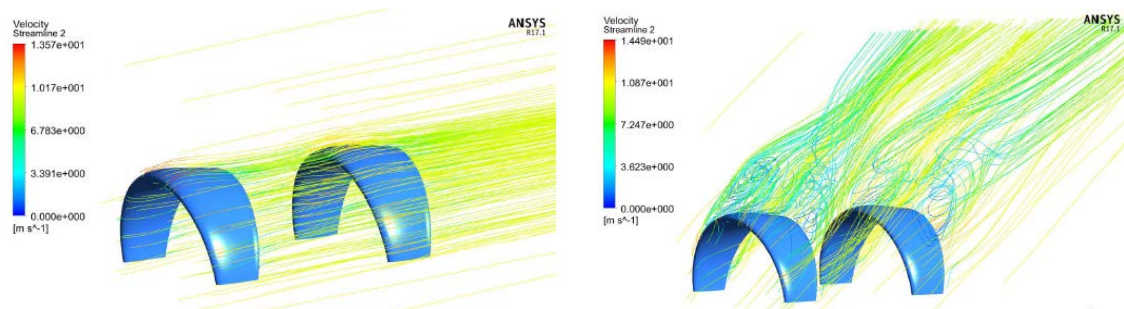
*Figure 2: Comparison of Experimental and CFD results*

## 2.Experimental Validation

In order to provide validation of the CFD simulation, a direct comparison was made between the simulated results, dimensionless analysis and the experimental results obtained via low-speed wind tunnel testing (Figure 2). It can be seen from Figure 2 that the experimental lift and drag results obtained via the low-speed wind tunnel, are slightly increased compared to the simulated CFD results. The deviation in the CFD results from the experimental results can be explained by CFD model is based on the assumption of the air entering the boundary is uniform, with constant air density and perfectly smooth kite surface. However, in the experimental setup, the air density is not uniform, and the 3D models have minor roughness. Although the experimental lift and drag coefficient curve shows slightly higher values than the CFD results, the curve is still a close match i.e. stall occurs at an equivalent angle of attack of  $40^\circ$  with a percentage error of 10% suggesting the CFD model can be used for further analysis.

### 3.Results and discussion

While in operation for power generation, it is likely that the power kites will be operating in close proximity. A single kite can create a large area of turbulent flow, which can impact the efficiency of a kite operating nearby. To find the ideal distance that the kites can operate safely, two kites were simulated by varying horizontal separation distances from 200 mm, 400 mm to 550 mm and vertical separation distance from 150 mm, 0 mm to -150 mm. It is to be noted that the simulation is scaled using a 1:35.5 model; the distance separating the models can be scaled to appropriate the actual position when in industrial use.



*Figure 3: Comparison of velocity streamline plot for horizontal separation distance 400 mm and 200 mm*

Figure 3 shows the comparison of velocity streamline contour plots with horizontal distance 400 mm and 200 mm respectively, and Figure 4(a) shows both the comparison of the Lift and Drag Coefficients of the three different horizontal kite separation distance simulations. When comparing the Drag Coefficients, it can be concluded that the drag is not a significant factor in the choice of arrangement to be carried forward, the coefficient of lift, however, does vary, meaning a setup can be chosen from this. In comparison, the Lift produced by each wing is very similar throughout each simulation, with the greatest variation occurring at an AoA of both  $15^\circ$  and  $25^\circ$ . At these deviations, the lowest Lift Coefficient, and therefore the lowest efficiency, is seen to be at a horizontal displacement of 400 mm, in this setup, the leading kite produces the lowest lift coefficient out of the three simulations, with the rear producing the 2nd lowest. From these results, it can be concluded that the horizontal displacement of 400 mm creates the least efficient setup.

Figure 4(b) shows both the comparison of the Lift and Drag Coefficients of the three different vertical kite separation distance simulations. From this figure, the drag coefficient is almost identical for each setup, the only variation being at  $15^\circ$  AoA for the trailing kite located above the leading. The lift coefficient for each kite working close together is greater than that of the single for most AoA. The greatest lift coefficient, produced at  $40^\circ$ , is created by the leading

kites in both setups, the trailing kite located 150 mm below generally produces a more significant lift over a wider AoA, and therefore this setup will be able to obtain more energy out of the wind, thus obtaining a more efficient setup for power generation.

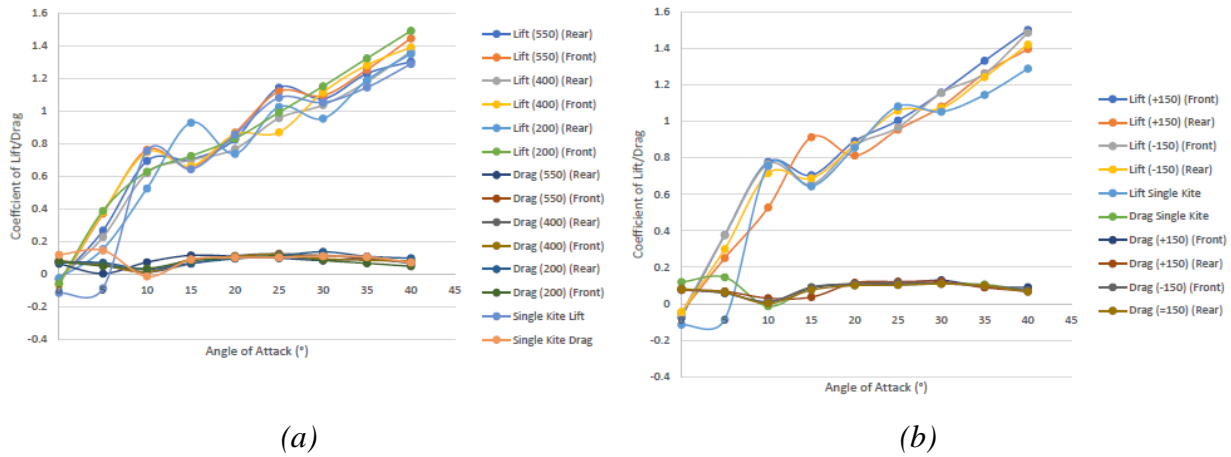


Figure 4: Comparison of lift and drag coefficients for different (a) horizontal separation distances (b) vertical separation distances

From the results above, it can be concluded that the leading kite will always produce a lift coefficient greater than that of the trailing kite, this lift coefficient, however, can vary. The rear kite at a horizontal displacement of 200 mm only generally produces the lowest lift coefficient out of the four kites, with the rear kite for the vertical setup producing a greater lift for a majority AoA. For the front kite, the efficiency of the two setups is very close for several AoA, with generally the same lift coefficient produced. To conclude, at these specific conditions, the most energy efficient setup produced by the power kites is at a horizontal displacement of 400 mm (14.2 m), with a vertical displacement of -150 mm (5.325 m).

#### 4. Effect of Surface Roughness

All previous CFD results assume the kites surface to be perfectly smooth. In reality, the surface roughness is present and play an essential role. Therefore simulating a surface roughness on the kite is an essential factor. The roughness height and roughness constant used in the present student is 0.001 m and 0.5. The results obtained are shown in Figure 5. Figure 5 compares the drag of a smooth kite with that of the rough; it is shown that the rough kite has a larger drag coefficient and a lower lift. A larger drag is expected, as the kite surface imposes a larger resistive force to the air flowing over it, which in turn reduces the wind speed over the kite surface and reduces the lift.

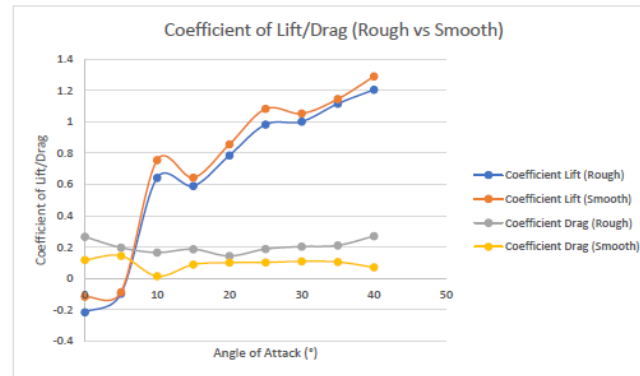


Figure 5: Comparison of lift and drag coefficients for smooth and rough aerofoil

## Conclusion

This paper aimed to optimise a power kite suitable for renewable power generation; one of the main topics discussed was the operating distance between kites and how it impacts the efficiency of each kite. Generally, the leading kite is not impacted when working in close vicinity to other kites, the trailing one, however, was subject to significant variations in lift and drag coefficient. These results suggested that the optimal positions for the rear kite are either to be directly behind the leading kite, a large distance between the two (at least 15 m) or for the trailing kite to operate below the leading. Due to the risk of kite tethers wrapping around each other if operating in close proximity, the ideal choice is operating the kites at minimum 15 m horizontal separation, with the trailing kite operating below the leading, removing the potential for this kite to operate in the wake turbulence of the first. As the wind speed is generally very low at low altitudes, it is recommended to use a symmetrical aerofoil for the kite design, as these tend to produce greater lift with low Reynolds airflow.

## References

1. Blanco MI (2009) The economics of wind energy. Renewable and sustainable energy reviews, 13(6-7), pp.1372-1382
2. Zhang H (2013) Kite modelling for higher altitude wind energy. Energy and Power Engineering, 5(07), p.481.
3. Canale M, Fagiano L, Milanese M (2009) High altitude wind energy generation using controlled power kites. IEEE Transactions on Control Systems Technology. 18(2), pp. 279-93
4. Gerakopulos R, Boutilier M and Yarusevych S (2010) Aerodynamic characterization of a NACA 0018 airfoil at low Reynolds numbers. In 40th Fluid dynamics conference and Exhibit (p. 4629)
5. Bauer F, Kennel RM, Hackl CM, Campagnolo F, Patt M and Schmehl R (2018) Drag power kite with very high lift coefficient. Renewable energy, 118, pp.290-305